Layout Considerations

General Application Note 101

Power Supply Layout for Printed Wiring Boards in Distributed Power Systems

Most modern telecommunications applications that use dc-dc converters at the PWB level use a power distribution scheme commonly referred to as *DC distributed power architecture*. Refer to Appendix 1 for a brief system level description of DC distributed power systems.

Within this framework, specific telecommunications building blocks are usually designed on separate PWBs. These are commonly referred to as circuit packs, line cards, or application cards, and are capable of being plugged in to the system backplane via edge connectors.

The design of the board level power supplies for these circuit packs is critical to the successful and reliable operation of the entire telecommunications system. Optimized PWB layout of the power supply section and use of the associated power supply components within their intended limits will help greatly in accomplishing this goal.

The typical PWB power supply consists of several functional blocks, which include:

- Input filter and other filtering components to control system noise
- Hot swap circuitry usually incorporating a soft start feature to limit the board inrush current
- One or more dc-dc converters (and a few associated external components) to convert the distribution bus level to the various requisite voltages needed to operate specific circuitry on the board
- PWB layout - Sizing and routing of the PWB traces used to interconnect these items, including the traces from the output of the converters feeding the loads can also be considered part of the power supply design.

The following sections describe these functional blocks in somewhat more detail.

Input Filtering

Noise and EMC

Large electronic systems create, emit and are generally susceptible to the effects of electronic noise. Additionally, noise commonly labeled as electromagnetic interference (EMI), appears in both conducted and radiated forms and can be caused by many sources both internal and external to a given system. An in-depth discussion and understanding of noise in electronic systems, its sources and the effect on electronic circuits, is beyond the scope of this paper, and the reader is referred to the many texts available on the subject.

Control of EMI is accomplished by adopting the tenets of electromagnetic compatibility (EMC), defined by the IEC as the ability of a device or system to operate satisfactorily in it's intended environment without introducing intolerable electromagnetic disturbances to anything in that environment. For a given device, there are several terms associated with the term EMC:

- *Emissions* - refers to the level of EMI created by and emanating from a device to its environment. If a device is said to meet EMC, it implies emissions (EMI) are controlled to a specified acceptable level.
- *Susceptibility* - refers to the level of EMI emanating from the environment into a device and the degree to which this noise affects the operation of the device. If a device is said to meet *Immunity* requirements it implies it is not susceptible to incoming EMI to a specified acceptable level.

The level of EMI under which a device is considered to meet EMC is primarily controlled by regulatory statute. In general, telecommunications systems must meet requirements for radiated and conducted EMI per EN55022 (Europe and other parts of the world) and/or FCC part 15J and 47CFR, part 15B (USA). Further, in many areas of the world, there are mandatory immunity requirements. For example, the parent standard EN61000 includes a number of standards covering all aspects of product immunity. Other standards of significance include those from the European Telecommunications Standards Institute (ETSI). In particular ETS300 386, parts 1 and 2, which cover similar areas as the standards listed above, but do contain some differences.

The primary reason for input power filters in systems is to provide the necessary circuitry to insure a product meets conducted EMC standards, although design of such a filter can also impact a product's radiated emissions as well.

Input Decoupling

PWB level filtering usually starts with common-mode decoupling of the dc distribution bus. This can be easily done by connecting small ceramic capacitors (typically 0.033-0.047μF) from each side of the dc-distribution bus to a frame ground immediately after the card backplane connector. These components provide an effective common-mode AC ground reference for both power lines to frame ground, and preclude a significant amount of CM noise from entering or exiting the PWB. The voltage rating of these capacitors must be sized to meet any safety regulatory requirements, and must withstand any system hi-pot or dielectric strength test requirement between line and ground imposed by the standards. Typical values of 1500-2500V are not uncommon. These capacitors are commonly referred to as “Y-caps”, a term borrowed from the AC-DC power supply world, which refers to the connection of an EMI capacitor from the AC-mains to earth or safety ground.

Depending upon the system architecture, additional differential capacitance (so-called “X-caps”) may also be installed across the dc-distribution bus at this point. Typical values are 1-4.7μF, but the level of capacitance added to the board at this interface depends upon:

- The impedance of the dc-distribution bus looking towards the backplane (more inductive in nature generally requires more capacitance.
- Whether the system backplane already has such capacitance.
- The level of inrush current that can be tolerated through the backplane/PWB connector.

Power Filtering

Following input decoupling, typically a power filter with bi-directional characteristics is placed in the path of the dc-distribution bus to perform two functions.

- Attenuate noise generated by the on-board power converters (conducted emissions) to prevent contamination of the dc-distribution bus
- Attenuate noise emanating from the dc distribution bus to a level that precludes damage to other power supply components and in some cases will guarantee continued operation of the supply and the circuit pack during certain noise events (conducted immunity).

These requirements must be met for both common-mode and differential mode noise sources. As a result, a typical power filter consists of several stages and commonly includes components to address both types of noise sources.

The di/dt, Inc. **F4804 Input Power Filter** provides filtering for any converter and is ideal for use with any converter series manufactured by di/dt, Inc. It consists of several stages of capacitive differential attenuation, a series differential choke, and a common mode choke. See the F4804 datasheet for additional details on this product at (www.didt.com)

In addition to the filter’s tuning being appropriate to provide adequate attenuation over the desired frequency spectrum, it is very important to keep the dc current passing through the filter within the published dc rating; to do otherwise invites dc saturation of the differential choke and an attendant increase in board level noise levels.

Transient Suppression

Also included within the input filter section is a device to suppress voltage transients that may be seen at the input of the circuit pack to a level that can be tolerated by the rest of the PWB circuitry. Commonly called a Transient Voltage Suppressor (TVS), it is realized by a very fast acting avalanche diode, which provides sub-nanosecond turn-on and controlled voltage clamping. For example, see www.transorb.com. The TVS provides a level of immunity to circuitry otherwise susceptible to damage or improper operation when excited with high-voltage transients. If properly chosen this device can facilitate continued operation of a power supply and the associated circuit pack under conditions where fast high-voltage transients are present on the PWB input. For a 48V system, the device is usually chosen to limit the bus voltage to 100V or less.

Hot Swap Circuitry

Hot-swap circuitry allows a circuit card to be safely connected (“hot plugged”) to a live backplane without causing a significant transient or “glitch” on the dc-distribution bus. Transients of this nature can cause the shut down of adjacent circuit packs and even reset of an entire system.

Hot swap circuitry designs in IC packages are available from several manufacturers (e.g., www.summitmicro.com or www.linear-tech.com , and can operated directly from the dc-distribution bus, and provide a simple hot-swap solution by eliminating most external components except for the external control device, usually a MOSFET. When using hot-swap ICs in a board design, the use of a transient protection device as described in the last section is an absolute requirement for a reliable design.

DC-DC Converters

di/dt, Inc. designs and manufactures a number of models of dc-dc converters, to include quarter brick and eighth brick converters with single outputs and quarter brick converters with dual output capability. All dc-dc converters require a certain number of external components. Some are used to control the various features of the converter; some are used to provide additional control of EMI. A brief description and layout for these components follows.

DC-DC Converter Features

The operation of all features of di/dt, Inc designed converters is described fully in the individual datasheets available on the Internet at www.didt.com.

- Remote ON/OFF – The ON/OFF pin is used for on/off control of the converter by a low-level system signal. This pin is referenced to Vin(-) and connecting the ON/OFF pin to the Vin(-) will enable or disable the converter depending upon the logic option chosen for a given converter (see datasheet for details). The control device for this function is typically an opto-isolator or a FET switch, and it should be located close to the ON/OFF and Vin(-) pins as possible. To minimize noise pick-up, the traces should be as short as possible. If a single device is used to control more than one converter, route the traces as close as possible to one another to minimize loop area and possible noise pick-up. To minimize noise pick-up in new designs, di/dt suggests the use of the negative logic version where possible, because the ON/OFF pin will be pulled down to the Vin(-) level during operation.

  Using the same reasoning, if the ON/OFF function is not to be utilized in a given design, it is preferable to specify the negative logic device and provide a hard-wire (trace) connection between the ON/OFF pin and the Vin(-) pin.
- Remote Output Voltage Sensing – The SENSE+ and SENSE- pins are the voltage feedback connection to the converter's output voltage regulator. In any application, these pins must always be connected to their associated output pins, Vout(+) and Vout(-), respectively. These leads can also be connected to the Vo(+) and Vo(-) traces at some distance away from the converter, and doing so will cause these points to become the output voltage reference location for the dc-output bus. These traces can be very small, as they do not any carry any significant current. When remotely connecting, the two traces should be routed very closely to one another to minimize the enclosed loop area to minimize noise pick-up.
- Remote Output Voltage Trim – External resistors can be used to adjust up or down the nominal output voltage of a given converter. The resistor value and connection for the desired trim (+ or -) depend upon the converter used. Please consult the specific datasheet for this information. In general, a trim resistor must be connected
  - between the trim pin and either the SENSE+ or SENSE- pins (single output converters) or
  - between the trim pin and either the Vo(+) or Vo(-) pins (dual output converters).

  Care should be taken to minimize the trace lengths and enclosed loop area for any of these connections.

DC-DC Converter Local EMI Control

Additional components are typically added around the converter, to provide local control of EMI. These items are

- Converter Input Capacitor – This capacitor connects across the input terminals of the converter and provides two functions.
  1. Compensate for an input bus with highly inductive source impedance
  2. Provide some additional EMI protection.

  To accommodate both purposes, this capacitance is comprised of two separate capacitors, a bulk capacitor, usually an electrolytic (ESR<1Ω), in parallel with a small ceramic capacitor for high frequency EMI control. A single ceramic capacitor of 33-100 uF can also be used, but provision should be made to add a small (typically <1Ω) series damping resistor, if necessary.

  As noted previously, all conducted EMI measurements to EN55022 must be made using the line impedance stabilization networks (LISN) specified by CISPR Publication 16. These are the same LISNs used as measuring ports for equipment operating from the AC mains. Each LISN has several hundred micro-Henries of series inductance two are used (one per input line), hence adding an additional 100 µH to the input impedance seen by the converter. This extra Inductance can cause EMI readings significantly worse than what would be seen in an actual application where these inductors would not be present. As a result, using the larger sized input capacitor is a good practice.

In the case where designers specify input capacitors (2000-5000 µF) installed on the backplane directly after the main DC-input terminals of the system frame, the converter input capacitor can be small, typically in the range of 33-47 µF. When no system input capacitors are installed, the value of the converter input capacitor may need to be a low ESR electrolytic, 100 µF or larger.

- Common-mode Bypass Capacitors – In most cases for additional control of CM noise, small value (0.1-1.0 µF) capacitors are also connected from each power line [Vin(+), Vin(-), Vout(+), Vout(-)] to either a shield plane(s) or independently to frame ground. The same requirements for the capacitor voltage rating, as described in an earlier section on line decoupling capacitors also apply to these capacitors.

PWB Layout

As important as the choice of active and passive devices used to perform power and end application functionality, the design of the PWB to which these items are attached and interconnected is at least as important, and with regard to control of both radiated and conducted emissions, probably most important.

Because parasitic impedances are non-trivial at frequencies in the audio region and above, what appears as a simple trace becomes a complex path - much more than just a low resistance interconnect that would be seen under DC measurements. Due to these parasitic impedances – both capacitive and inductive in nature, the PWB can become the designer's worst EMI nightmare if not properly designed.

For the power supply area the following design practices are recommended. Most telecom and datacom circuit packs are constructed using multiplayer PWBs. The following comments assume this construction. As a cautionary note, PWBs must meet certain safety requirements with respect to dielectric strength between the various planes. Be sure these requirements are understood before designing any PWB.

- Input power traces to the input power filter – when a multiplayer board is being used, run the positive and negative bus traces on multiple separated layers of the PWB. Create the power input traces as large conductive planes. The power traces on these power planes, should mirror one another in size and shape as much as possible.
- System Frame Ground – if the circuit pack has a backplane connection to the system frame ground, it is good practice to connect this pin to a separate board layer and where possible, multiple layers. Each layer should be where possible a full conductive plane. The exception to this rule would be for keep out areas where application requirements of the specific devices indicate otherwise.

Recommended Layouts

Figure 1 provides a view of the single converter safety creepage boundary. Where necessary for safety considerations, care must be taken to insure required creepage boundary once converter has been placed. This is typically not an issue if the top surface of the application PWB is kept clear of any copper.

Figure 2 provides some design information for layout of SM pads and the vias to connect these pads to inner power planes.

Figures 3-9 provide detailed board layouts for all currently available didt converters and the F4804 filter.

Note: the drawings shown for the through-hole converters show only the component side geometry and do not provide sizes of the interconnecting solder pad located on the solder side of the board. This detail is left to the discretion of the PWB designer to apply any local standards as required.

Radiated Emissions

From the systems perspective, far-field radiated noise is the primary concern of the regulatory agency specifications. The level of this noise is a function of the cabinet geometry, the integrity of shielding provided as a part of the cabinet, and ability of the many radiated energy sources within the system to couple to the structure.

Layout Considerations

General Application Note 101

All boards to include the separate circuit packs and the system back planes have near field and far field components of radiated emissions. To minimize these at the board level, care should be taken to avoid unintentional radiators or antennas by judicious layout of critical areas of the circuit pack. These critical areas are those associated with high frequency signals. Details of methods to minimize radiated emissions can be found in a number of readily available texts and publications. All di/dt converters pass EN 55022 Class A Radiated requirements as a standalone device (on a PWB with an F4804 filter.)

For power converters in particular, use of bypass capacitors located close to the converter, from each power connection to a frame ground or floating shield plane can indirectly control the level of near-field radiated emissions. Because these capacitors provide a low impedance path for conducted emissions, this energy has a much lower capability of being transformed into a radiated form. Examples of the use of a shield plane and other practical layout information can be found in di/dt **Application Note 100**.

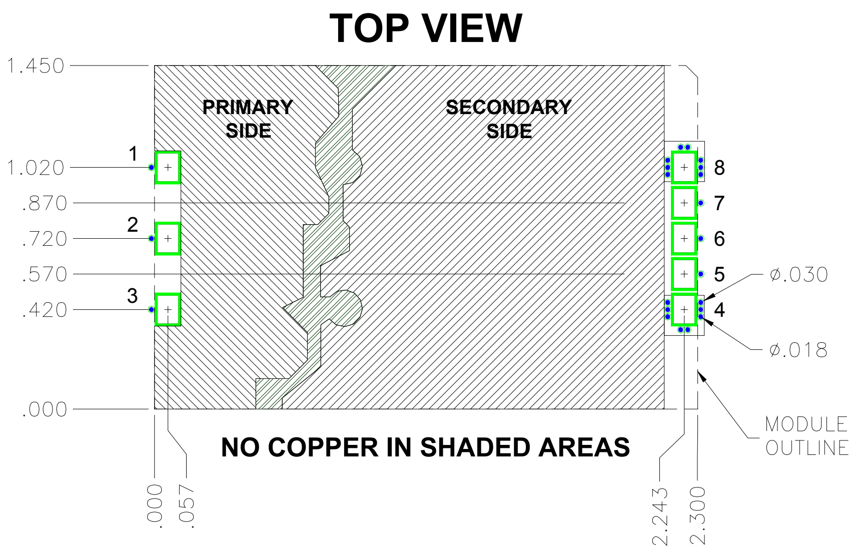

Figure 1: Single output converter primary to secondary creepage boundary

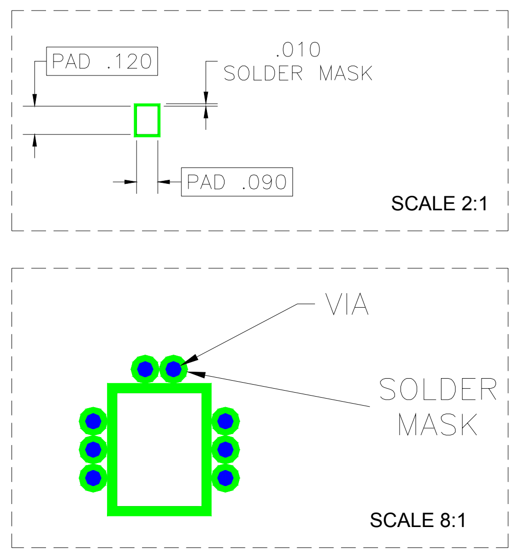

Figure 2: Suggested via layout for output power pins. All other pins have one via.

Figure 1: Details of primary/secondary keep-out area. The irregular shaped shaded area between the primary and secondary sides represents the converter creepage barrier. Care must be taken to insure necessary creepage boundary once converter has been placed. This is typically no an issue if the top surface of the application PWB is kept clear of any copper. Top view shown, looking through converter

Figure 2: Suggested via layout for SM pads for single output converters (SQ24S, SQ48S series). Vias are used to interconnect the SM pad to internal power planes. The number of 0.018” (0.457mm) diameter vias is based on a maximum of 3.75A/via; these values can be used for design of other SM layouts.

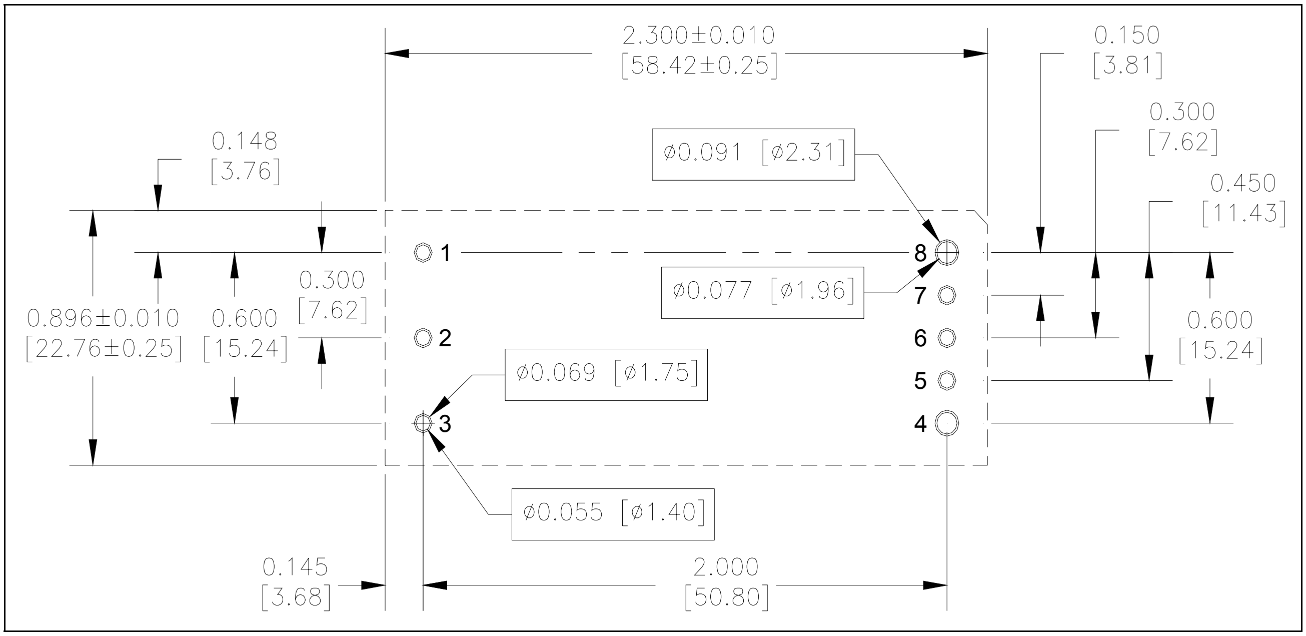

Figure 3: Suggested Layout for single output through-hole converters (SQ24T, SQ48T series)
Top view shown, looking through converter

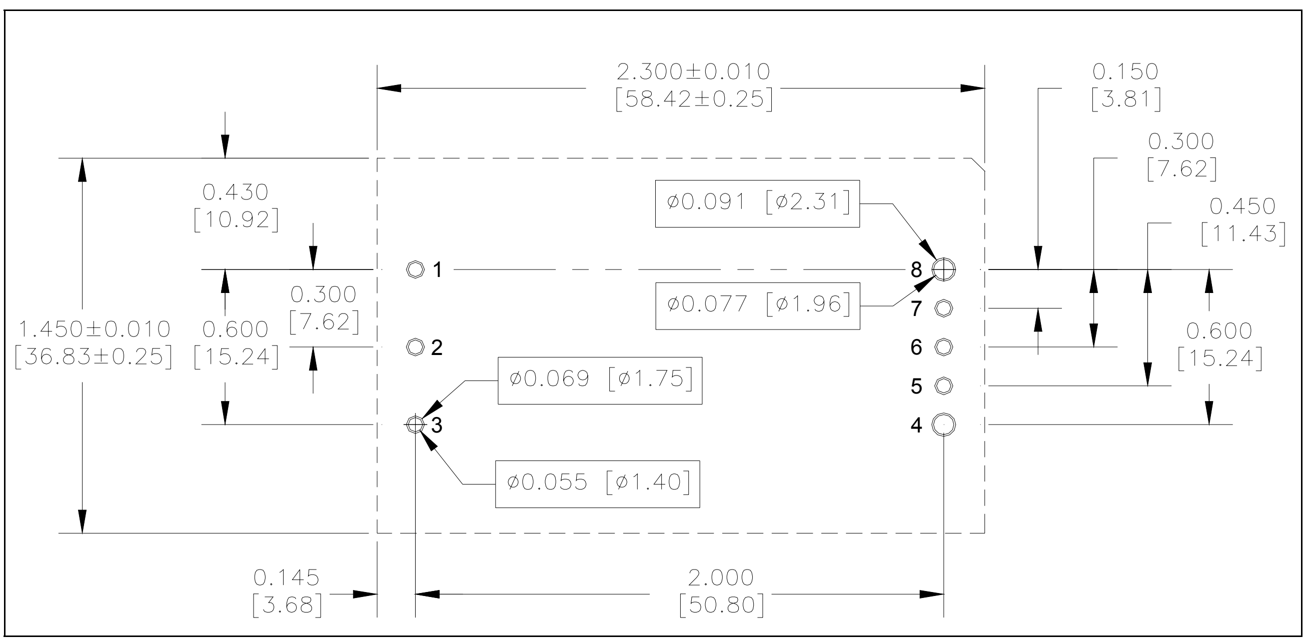

Figure 4: Suggested Layout for single output through-hole converters (Q24T, Q48T series)
Top view shown, looking through converter

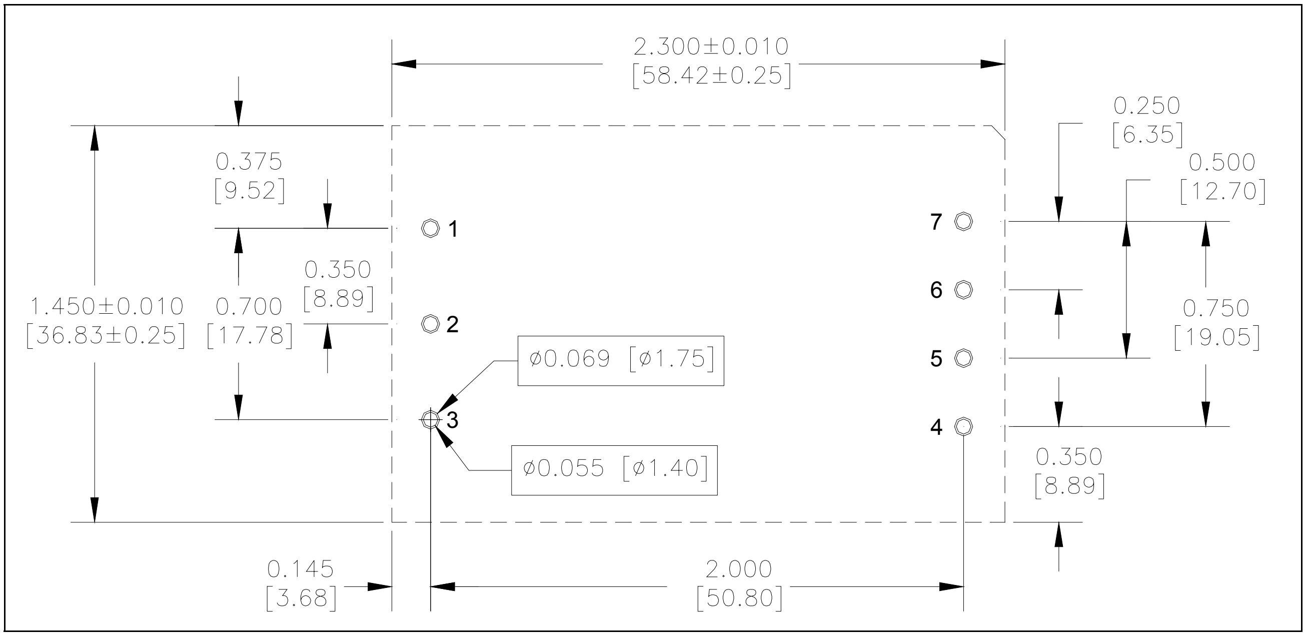

Figure 5: Suggested Layout for dual output through-hole converters (QD24T, QD48T series)
Top view shown, looking through converter

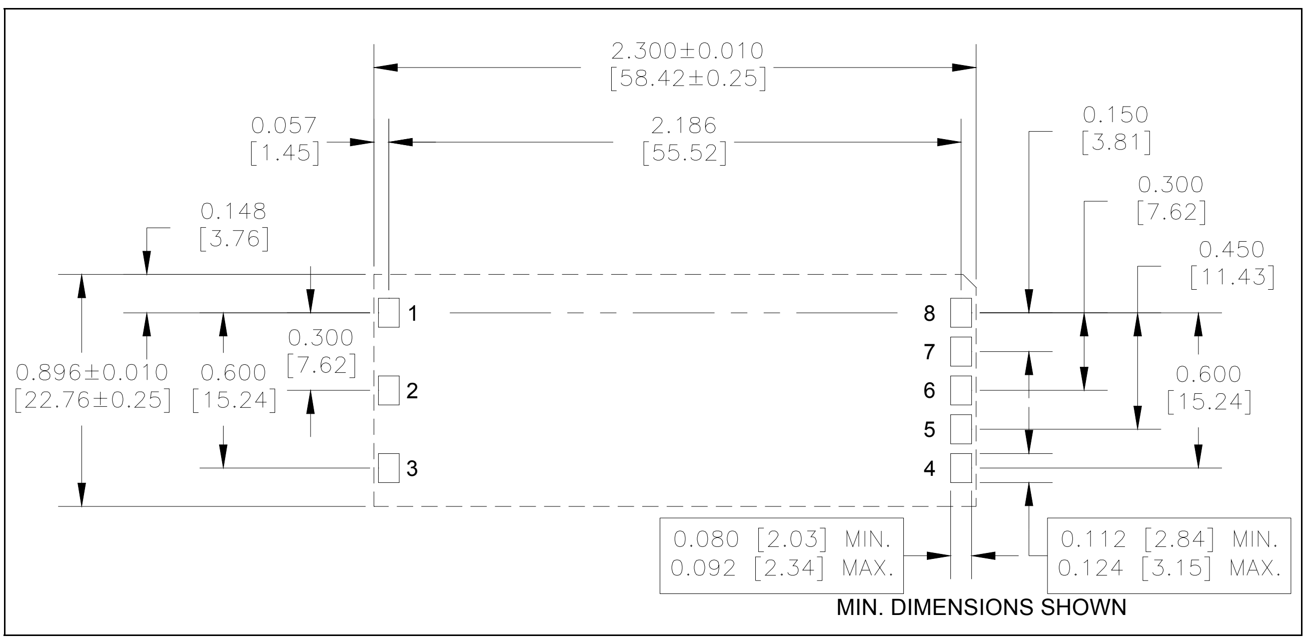

Figure 6: Suggested Layout for single output surface mount converters (SQ24S, SQ48S series)
Top view shown, looking through converter

Layout Considerations

General Application Note 101

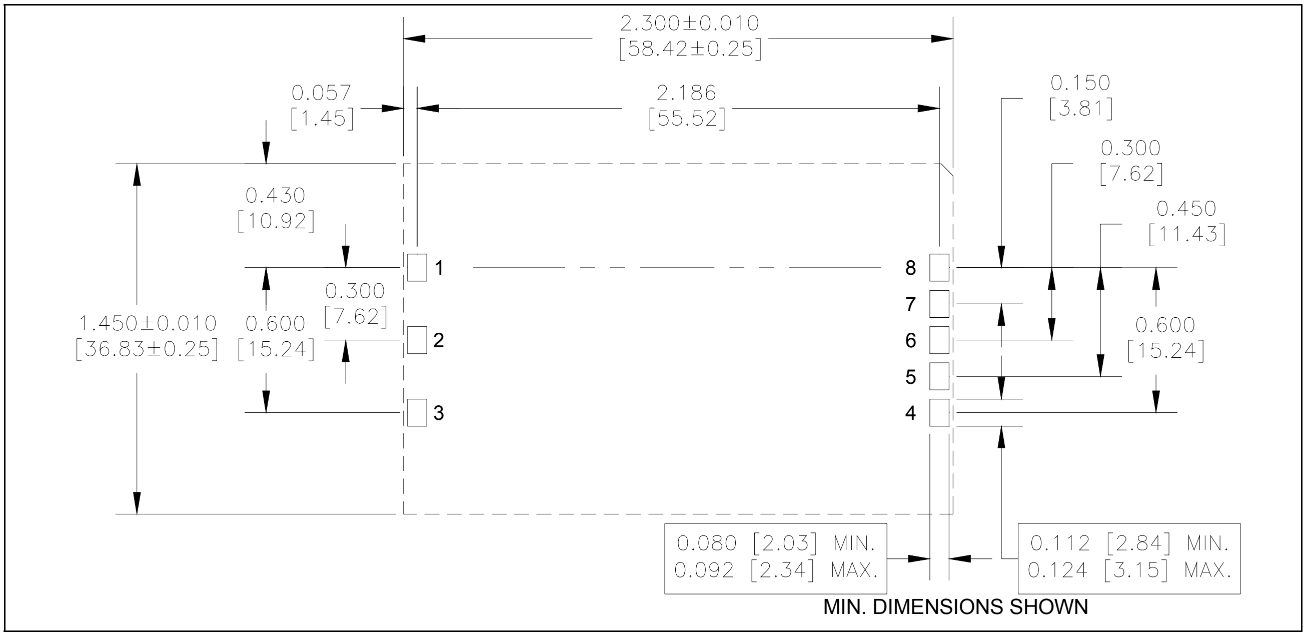

Figure 7: Suggested Layout for single output surface mount quarter-brick converters (Q24S, Q48S series)
Top view shown, looking through converter

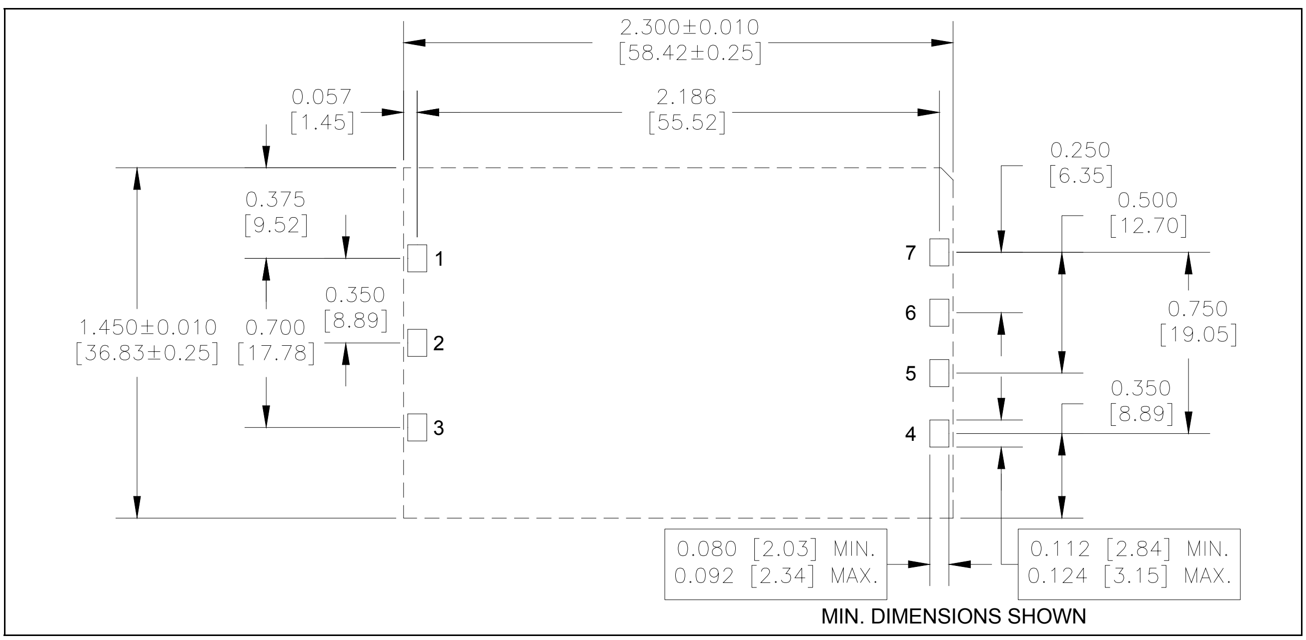

Figure 8: Suggested Layout for dual output surface mount quarter-brick converters (QD24S, QD48S series)
Top view shown, looking through converter

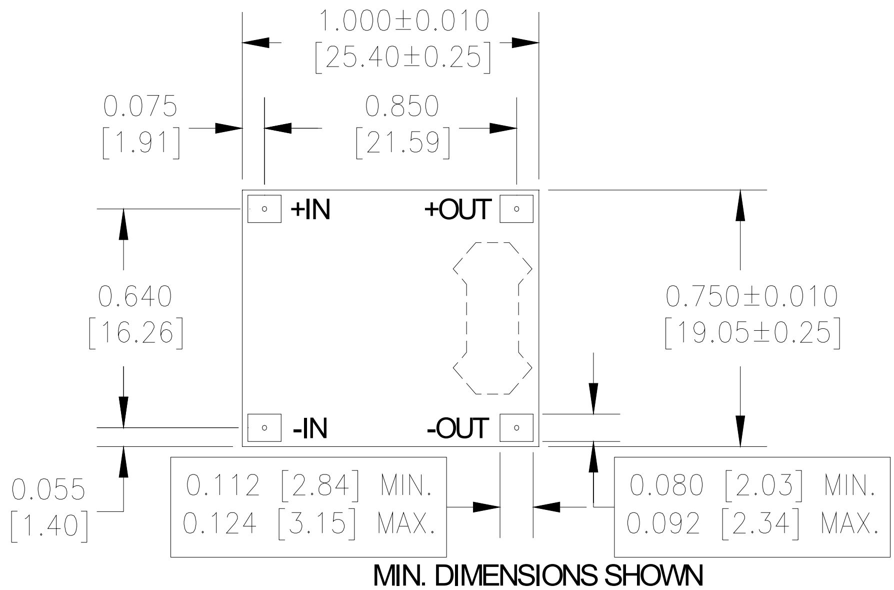

Figure 9: Suggested Layout for filter module (F4804)
Top view shown, looking through module

Appendix 1

Distributed Power Systems

Most modern telecommunications applications that utilize dc-dc converters utilize some form of a distributed DC power system. This type of power distribution consists of ("front end") rectification equipment used to convert incoming AC mains voltages to a common intersystem DC distribution bus voltage (typically +12V, +24V, -48V and in Europe, -60V), one or more card cages or shelves which house a number of printed wiring boards (PWBs) and provide electrical connection of the cards to the distribution bus via the shelf back planes. This power system design is used because of the ease of providing redundancy (e.g., battery back-up or dual dc-plant feeds) to minimize transmission losses and guarantee tight regulation at the point of load (board level).

For high availability systems the distribution bus will have dual front ends (OR-ed feeds) and battery back-up capability. When so equipped, the actual bus level will be somewhat higher to accommodate the "float voltage" of the batteries. For example the normal operating bus voltage of a "-48 V system" when equipped with battery back-up will operate at approximately –54 V, due to the extra voltage required to keep the batteries constantly charged.

When a condition occurs which disables the AC mains, the bus voltage will drop from the system float level to the battery terminal voltage (-48V, in the example above), and begin discharging through the load. The battery back-up system will maintain the load until such time as the AC mains or some other AC power source such as a motor driven alternator comes on line and begins recharging the batteries, or when the bus voltage falls to a system defined point (typically 38-39 V) where the batteries are automatically disconnected to prevent complete discharge and likely battery failure.

This DC bus voltage is used to distribute power to all parts of the system, by connecting it to the system shelves or PWB frames. Each shelf typically holds 3-18 PWBs, depending upon the size and intended application of the system. It is primarily at the PWB level where DC-

Layout Considerations

General Application Note 101

DC converters are used to convert the high DC distribution bus voltage level to the lower voltages (typically 5.0V, 3.3V, 2.5V, 2.0V, 1.8V, 1.5V, 1.2V) needed to operate the electronics circuitry used to implement the telecommunications function of a specific card.

For additional information, please contact

di/dt, Inc.

1822 Aston Avenue •• Carlsbad, CA •• 92008 •• USA

USA Toll Free 866-WOW-didt (969-3438)

www.didt.com •• support@didt.com